AIAA-2001-1602

IMPACT TEST AND SIMULATION OF ENERGY ABSORBING CONCEPTS FOR EARTH ENTRY VEHICLES

Marcus D. Billings*
The George Washington University, Joint Institute for Advancement of Flight Sciences
Hampton, VA

Edwin L. Fasanella†
US Army Research Laboratory, Vehicle Technology Directorate
Hampton, VA

Sotiris Kellas‡
Veridian Systems
Yorktown, VA

Abstract

Nonlinear dynamic finite element simulations have been performed to aid in the design of an energy absorbing concept for a highly reliable passive Earth Entry Vehicle (EEV) that will directly impact the Earth without a parachute. EEV's are designed to return materials from asteroids, comets, or planets for laboratory analysis on Earth. The EEV concept uses an energy absorbing cellular structure designed to contain and limit the acceleration of space exploration samples during Earth impact. The spherical shaped cellular structure is composed of solid hexagonal and pentagonal foam-filled cells with hybrid graphite-epoxy/Kevlar cell walls. Space samples fit inside a smaller sphere at the center of the EEV's cellular structure. Comparisons of analytical predictions using MSC.Dytran with test results obtained from impact tests performed at NASA Langley Research Center were made for three impact velocities ranging from 32 to 40 m/s. Acceleration and deformation results compared well with the test results. These finite element models will be useful for parametric studies of off-nominal impact conditions.

Introduction

An Earth Entry Vehicle (EEV) is designed to return materials from asteroids, comets, or planets to Earth. A current concept for an EEV is a circular aeroshell structure approximately one meter in diameter with an energy absorbing impact sphere in the center of the vehicle. A simple, highly reliable, and cost-effective EEV would be a vehicle that has a direct entry to Earth and can withstand a terminal velocity land impact without a parachute[1]. Thus, an optimal impact surface for an EEV would be soft clay soil[2]. Design criteria for the EEV concept require that sample containment be assured with high levels of reliability. Thus, an energy absorbing impact sphere has been designed to limit the acceleration of the samples and to provide a high level of containment to increase reliability. A concept for the EEV and the system components is shown in Figure 1.

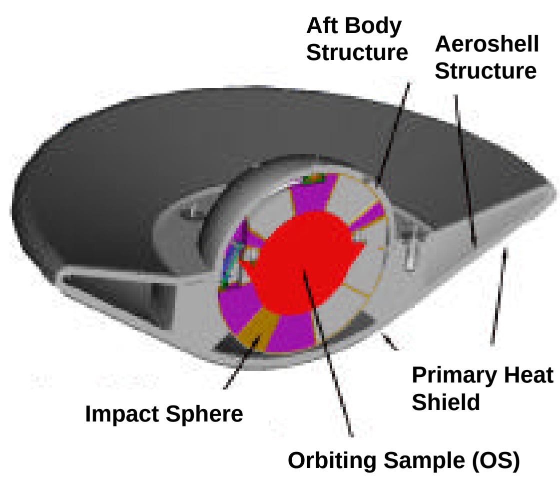

Figure 1 – Schematic of candidate Earth Entry Vehicle.

---

* Graduate Research Scholar Assistant, NASA Langley Research Center. Student Member AIAA, ASME.
† Structural Dynamics Branch, NASA Langley Research Center. Member AHS.
‡ Principal Engineer.

The current concept of the energy absorbing impact sphere is a composite cellular structure made with energy absorbing materials that limits the acceleration of the space samples and ensure containment. The cellular structure is composed of cells that are filled with energy absorbing foam and enclosed with hybrid composite cell walls. Energy absorbing materials include carbon foam, Kevlar, graphite, and hybrid Kevlar-graphite composites. Rock, soil, and atmospheric samples may be kept within the cellular structure in a sample container designated the Orbiting Sample (OS), as indicated in Figures 1 and 2.

The required reliability and containment assurance criteria are being addressed, in part, by performing nonlinear dynamic finite element simulations of the impact of the EEV's cellular structure onto a rigid surface. These simulations serve as an aid in the design and testing phases of the program. The purpose of the simulations is to calculate accelerations and deformations of the cellular structure and compare with test results. Once confidence in the model is achieved, off nominal impact conditions could be simulated that would otherwise prove to be difficult and costly to perform. These simulations are executed using the commercial nonlinear dynamic finite element code MSC.Dytran[3].

The objective of this paper is to describe the modeling and simulation of the EEV cellular structure and to show comparisons of test data with numerical results. Each model created for the analysis will be described and compared with test results from the corresponding impact test performed at the NASA Langley Research Center's Impact Dynamics Research Facility (IDRF).

Cellular Structure

A series of impact tests of the cellular structure specimens was conducted at the IDRF to evaluate and optimize energy absorption and to protect the OS. A design criterion requires that the peak acceleration of the OS upon impact be limited to 2,500 g's to preserve sample integrity and must not exceed 3,500 g's to assure containment[1]. Additionally, the cellular structure must not crush completely on impact. For each test, variations were made to cell wall thickness, foam density, or OS size and shape. The cross section of a typical impact test specimen is shown in Figure 2.

The current impact test specimen is a hemispherical-shaped structure with six pentagonal shaped cells, five hexagonal shaped cells, and ten hexagonal shaped cells on the equator that were cut to form the hemisphere. The bottom view of a typical cellular structure is shown in Figure 3, which illustrates how the individual cells are assembled to form the hemispherical cellular structure.

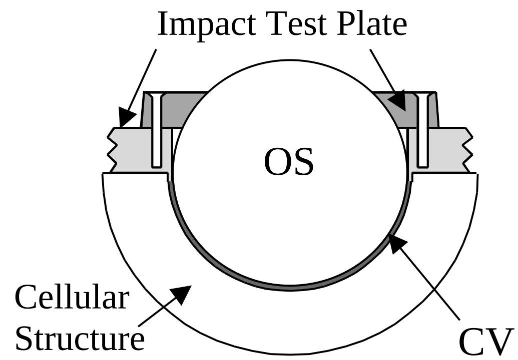

Figure 2 – Typical impact test specimen.

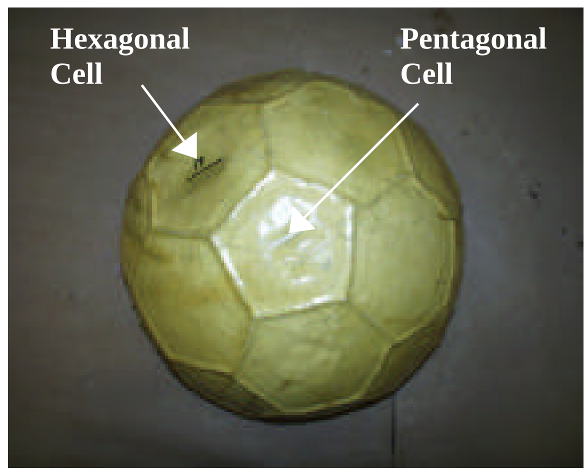

Figure 3 – Bottom view of cellular structure showing shape and geometry of the cells.

Each foam-filled cell has hybrid composite graphite-epoxy/Kevlar cell walls. The graphite-epoxy laminates provide stiffness and strength, while the Kevlar layers provide post-test structural integrity. Polyurethane foam was used initially to demonstrate the concept. The outer surface of the spherical cellular structure is wrapped in Kevlar, while the inner surface, which supports the OS, is constructed of a graphite-epoxy laminate. Between the larger, spherical, cellular structure and the OS are several layers of Kevlar fabric, which form the Containment Vessel (CV). Kevlar is laid inside the cellular structure to provide containment of the OS material and penetration resistance from foreign objects during impact. The OS is a sphere and will contain materials collected from exploration missions; i.e., rock, soil, and atmospheric samples. Advanced concepts include a titanium canister inside the spherical OS that holds the collected samples. Surrounding the titanium canister is foam that protects the canister over the course of the mission. An impact test plate has been added to the impact test specimen and represents the mass associated with the top half of the cellular structure, which was not constructed for the impact test program. The components of an impact test specimen can be viewed in Figure 4. A picture of an impact test specimen is shown in Figure 5.

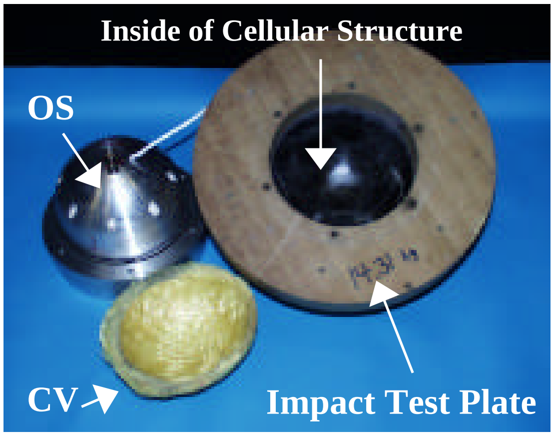

Figure 4 – Picture showing components of impact test specimen.

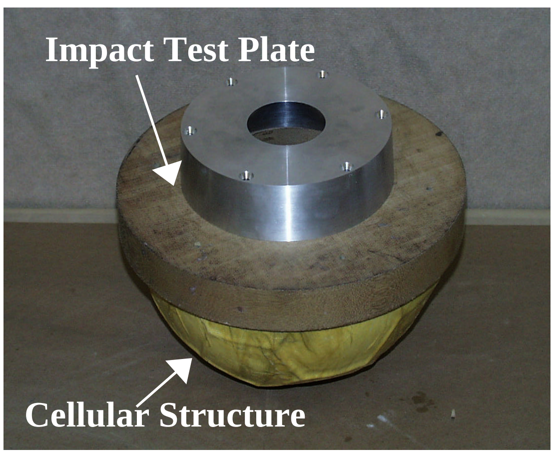

Figure 5 – Picture of typical impact test specimen with OS located within cellular structure.

Impact Tests at Impact Dynamics Research Facility

The initial tests of the cellular structure were simple instrumented drop tests from the 80 m high gantry structure at the IDRF[4]. Impact test specimens were released from the gantry, which generated impact velocities up to 33 m/s. A picture of the IDRF is shown in Figure 6. Recently, a bungee assisted accelerator system has been designed and constructed at the IDRF gantry to produce impacts up to approximately 40 m/s, the expected terminal velocity of this EEV concept. A diagram of the bungee accelerator is shown in Figure 7. In this paper, the modeling of three separate impact tests is described, illustrating development of the cellular structure concept. Table 1 outlines the three impact tests with pertinent test parameters provided.

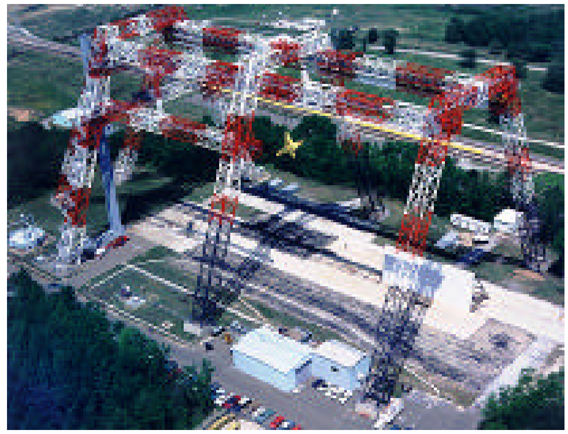

Figure 6 – IDRF at NASA Langley Research Center.

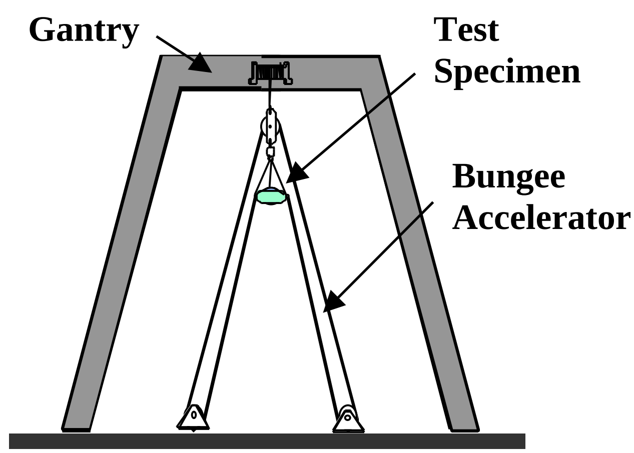

Figure 7 – Schematic drawing of bungee accelerator located under gantry structure.

Table 1 – Impact test parameters.

Test Specimen	Total Mass (kg)	Impact Velocity (m/s)
1	9.816	32.00
2	12.381	35.52
3	14.310	40.40

Nonlinear Dynamic Finite Element Code

The analysis was performed using the software code MSC.Dytran. MSC.Dytran is an explicit nonlinear transient dynamic finite element computer code used for analyzing solid components, structures, and fluids. The code offers a library of relatively simple structural elements including beams, shells, and solids for modeling complex structures. MSC.Dytran also offers a variety of material models for elastic, elastic-plastic, layered orthotropic, crushable foams, and soils; thereby allowing many engineering material systems to be analyzed. Several modeling options are available for contact, impact, and penetration including breakable joints and element erosion. Automatic time step control is provided once the user defines an acceptable initial time step. Archive files for post-processing deformed shapes and time history files must be requested by the user in advance. A restart capability is also provided.

Impact Simulation #1

The initial impact test was used to establish a method for dynamic testing of the cellular structure concept. The test results were also used to validate the design of the concept and to provide information for the design of future cellular structure concepts. The specimen was dropped from the gantry and had an impact speed of 32 m/s. The total mass of the cellular structure, CV, and OS was 9.816 kg. The modeled cellular structure had an outside diameter of 0.314 m and an inside diameter of 0.178 m. The thickness of the CV was approximately 0.0056 m. One accelerometer was mounted inside the impact test specimen to determine the OS response, and the second was mounted on top of the impact test plate. Additionally, a low-g accelerometer was used to determine impact speed. High-speed film and digital video were used to capture the impact event.

Finite Element Model

The major components of the impact test specimen were the cellular structure, the CV, and the OS. The components of the cellular structure were the energy absorbing foam, hybrid composite cell walls, the inner graphite-epoxy layer, and the outer Kevlar layer. The CV and OS were defined separately. The sample containing OS that is within the cellular structure was represented by rigid body shell elements located on top of the CV. Concentrated masses were applied at various locations across the top of the cellular structure to simulate the impact test plate, which represents the top half of the energy absorbing cellular structure. The impact surface was modeled with shell elements. Failure was not defined for any elements of the model. This constitutes the baseline finite element model for the impact analysis. The discretized model used to simulate the first impact test is shown in Figure 8. An exploded view of the impact model showing the major components is shown in Figure 9. Additionally, a cross-section of the discretized model is shown in Figure 10.

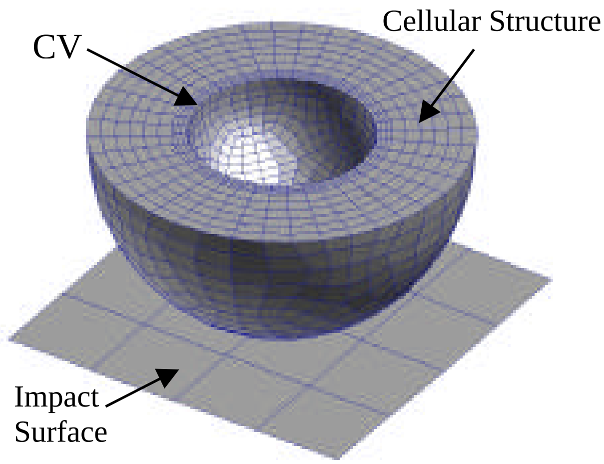

Figure 8 –Finite element model of test specimen.

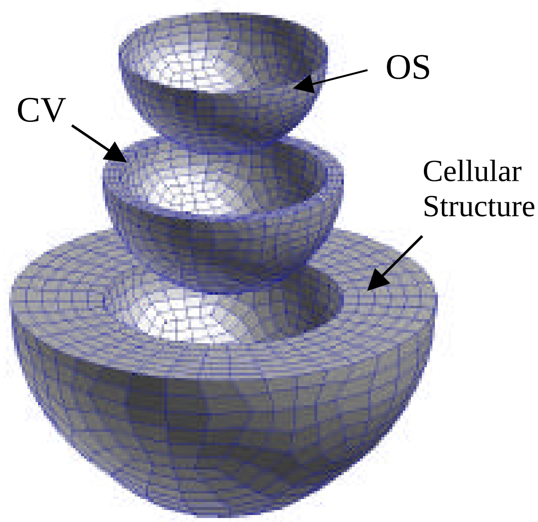

Figure 9 – Exploded view of analytic F.E. model used to simulate the impact test.

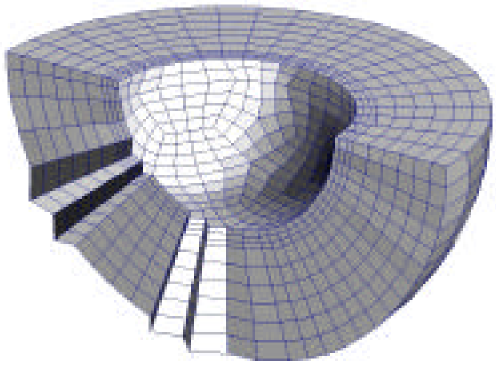

Figure 10 – Discretized cross section of test specimen.

The MSC.Dytran model contained 4,635 grid points and 6,616 elements. The model consisted of seven groups of elements, including the foam in each cell, the CV, the graphite-epoxy/Kevlar cell walls, the outer and inner surface of the cellular structure, the rigid body OS, and the impact surface.

The polyurethane foam was modeled with 2,640 8-node Lagrangian solid elements. The material model for these elements was an isotropic, elastoplastic material model with a bilinear stress strain curve and a yield stress defined to allow for plastic deformation. In the analysis, isotropic material properties were defined for the foam based on testing of foam core samples. The Young's modulus E was 1.982×10^7 Pa, and the yield stress was 1.034×10^6 Pa. The density ρ was 85.78 kg/m^3, and the Poisson's ratio ν was assumed to be 0.3.

The CV was represented by 1,320 8-node Lagrangian solid elements, with three elements defined through the thickness of the CV. For each solid element, there is one integration point located at the center of the element. For improved prediction of the stress distribution through the thickness of the CV, three elements are needed. The material model for these elements was an isotropic, elastoplastic model where the values were determined based on the properties of the Kevlar fabric. The Young's modulus E was 7×10^{10} Pa, the Poisson's ratio ν was 0.3, and a value of 3.44×10^7 Pa was used for the yield stress. The CV density was determined by dividing the mass of the CV by the calculated volume from the finite element code. This density ρ was found to be 383 kg/m^3.

To model the graphite-epoxy/Kevlar cell walls, 1,320 4-node Lagrangian shell elements were used with an isotropic, elastoplastic material with a shell thickness of 0.003048 m. In this analysis, the equivalent isotropic material properties for the cell walls were extracted from tensile test coupons, which represented the quasi-isotropic lay-up of the cell walls. The Young's modulus E was 1.379×10^{10} Pa and the yield stress was 1.379×10^8 Pa. The density ρ was 1,539 kg/m^3 and the Poisson's ratio ν was assumed to be 0.3.

The outer hemispherical surface of the cellular structure was wrapped with Kevlar sheets and was modeled as an isotropic, elastoplastic material with 440 4-node shell elements. For this set of elements, the defined thickness was measured to be 0.0005 m. The density ρ was 1,379 kg/m^3, the Young's modulus E was 6.895×10^9 Pa, the Poisson's ratio ν was 0.3, and the yield stress was 1.034×10^8 Pa.

The inner surface of the cellular structure was a laminate composite of graphite-epoxy and was represented by 440 4-node shell elements with an equivalent isotropic, elastoplastic material model. For the set of elements, the defined thickness of the shells was measured to be 0.000762 m. The density ρ was 1,550 kg/m^3, the Young's modulus E was 4.55×10^{10} Pa, the Poisson's ratio ν was 0.3, and the yield stress was 5.79×10^8 Pa.

The OS was modeled using 440 4-node Lagrangian shell elements located on the top surface of the CV. These elements were defined as a rigid body and represented the mass and inertia associated with the OS. In the impact test specimen, the OS was represented by ballast that filled the inner volume of the impact sphere. To represent the OS mass in the model, the rigid body shell elements were defined to have the center of gravity (CG) that matched the CG of the ballast with the proper moments of inertia. The mass of the OS was defined as 3.9 kg and was given an initial velocity of 32 m/s.

The impact surface was modeled using 16 4-node Lagrangian shell elements with a thickness of approximately 1 m. The material model for these elements was an isotropic model to represent the impact surface at the IDRF. The impact surface was represented as deformable to avoid numerical difficulties in the contact algorithm when a surface is defined explicitly as a rigid surface.

Included in the model were ten concentrated masses distributed around the top of the cellular structure. These masses represented the weight associated with the impact test plate, which represented the top half of the cellular structure. Each lumped mass was 0.440 kg resulting in 4.4 kg of total mass added to the system.

The contact defined between the cellular structure and the impact surface was modeled using the penalty method. In the analysis, shell elements from the modeled impact surface were the master surface, and all nodes of the cellular structure were defined as slave nodes. At the beginning of the analysis, the cellular structure was positioned approximately one millimeter above the impact surface. This method ensured no initial penetration of slave nodes into the master surface upon starting the simulation.

Numerical Results

All finite element simulations were performed using the MSC.Dytran finite element code on a Sun Ultra Enterprise 450 engineering workstation. The maximum time step for the impact simulation was 0.1593 microseconds. The simulated impact was run for 0.004 s to ensure that the complete acceleration pulse was captured. After this time, the cellular structure begins to rebound and the simulation was terminated. The finite element simulation took approximately three CPU hours to complete the simulation.

Data from the OS accelerometer was compared to the analytical acceleration response of the rigid body OS in Figure 11. The OS accelerometer data was filtered with a 2,500 Hz low-pass filter. The numerical simulation of the impact determined that the OS experienced a peak acceleration of approximately 2,900 g's. The measured peak acceleration of the OS was 2,560 g's, which was within the design limit. Although the analytical peak value was slightly higher, the overall shape and duration of the acceleration pulse compared favorably to the experimental acceleration pulse.

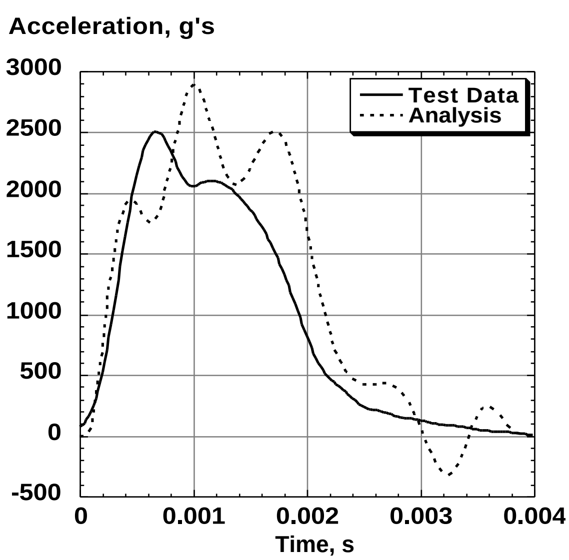

Figure 11 – Comparison of predicted results with experimental acceleration of the OS.

The finite element simulation of the impact event predicted a total crush stroke of 0.036 m, or 53% of the available crush distance of the model. The impact test showed that the cellular structure experienced a total crush stroke of 60%, or 0.041 m. These results indicate that the numerical model was slightly stiffer, and that more energy needed to be dissipated from the system during the simulated impact. However, the predicted maximum crush stoke was only 12% lower than the experiment. A deformed plot of the cellular structure is shown in Figure 12.

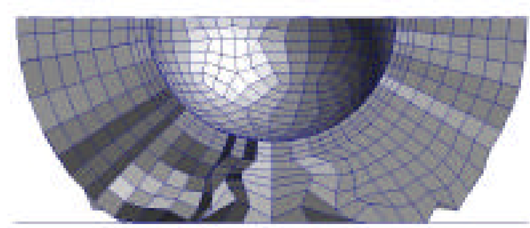

Figure 12 – Cross section of deformed impact structure at 0.002 s showing maximum crush.

Impact Simulation #2

The objective of the second impact test was to provide additional data for the design of the cellular structure at higher impact speeds more representative of the EEV's terminal velocity. In order to achieve the higher impact velocity, a bungee accelerator was installed under the gantry. The impact velocity of the second test specimen was 35.52 m/s. The impact test specimen was constructed using stronger, higher density polyurethane foam. The total mass of the cellular structure, CV and OS was 12.381 kg. The impact test specimen had an outside diameter of 0.314 m and an inside diameter of 0.178 m. The thickness of the CV was 0.0056 m. One accelerometer was located within the impact test specimen to capture the OS acceleration response. Another accelerometer was mounted on the impact test plate to measure the cellular structure response. A low-g accelerometer was used to determine the impact velocity. Digital video was also used to capture the impact event. For the impact test, the OS acceleration was completely lost due to a failure in the accelerometer-umbilical connection. The acceleration of the plate was captured over the first 0.0025 s, before the cable was also severed.

Finite Element Model

Changes were made to the baseline finite element model to represent the physical changes made to the impact test specimen. Otherwise, model discretization was unchanged. Different material models were used for the polyurethane foam and for the hybrid webs to more accurately model the increased crushing that occurred in this test. The polyurethane foam in the cells was modeled using the FOAM1 material model, which has a Poisson's ratio of effectively zero. The FOAM1 material model in MSC.Dytran was developed to represent foam crushing and is a more accurate representation than the elastoplastic material model used for the foam in simulation #1. A user-defined table was implemented in the FOAM1 model to represent the piecewise linear stress-strain curve acquired from a quasi-static crush test. Since the Poisson's ratio is effectively zero, only one other elastic constant was needed. For the polyurethane foam material, a bulk modulus of 8.21×10^6 Pa was used. The polyurethane foam had a density of 111.514 kg/m^3 and a yield stress of 1.319×10^6 Pa. Material failure was not defined for the foam elements.

Due to the higher impact velocity and increased deformation expected, the material model of the cell walls was modified to include a failure criterion. The failure model in the cell walls was based on a maximum plastic strain criterion of 20%. Thus, the elements in the cell walls will fail once a plastic strain of 20% is reached.

The overall mass for the second impact test specimen was increased. In the previous model, localized deformations associated with the lumped masses were observed around the top of the finite element model. In order to remove the localized deformation, the concentrated masses were distributed over 30 nodes around the top of the model. Each lumped mass was 0.214 kg, resulting in 6.415 kg of total mass added to the model.

Numerical Results
The simulation was executed for 0.004 s with the impact taking approximately 0.0035 s before the structure rebounded. This ensured that the simulation captured the entire acceleration pulse. The maximum time step for the simulation was 0.1593 microseconds. A total run time of approximately 3.5 CPU hours was required for the simulation.

The measured acceleration response of the plate was plotted and compared with analytical results from the finite element analysis in Figure 13. The plate acceleration data was filtered with a 5,000 Hz low-pass filter. The analysis calculated a peak acceleration of the cellular structure to be approximately 2,400 g's. The peak acceleration value of the test was 2,700 g's, a value that was 11% higher than the analytical peak. However, the analytical acceleration pulse shape closely matched the experimental acceleration pulse. Although the test data ends at 0.0025 s, the simulation was carried out to show the predicted response of the cellular structure through rebound.

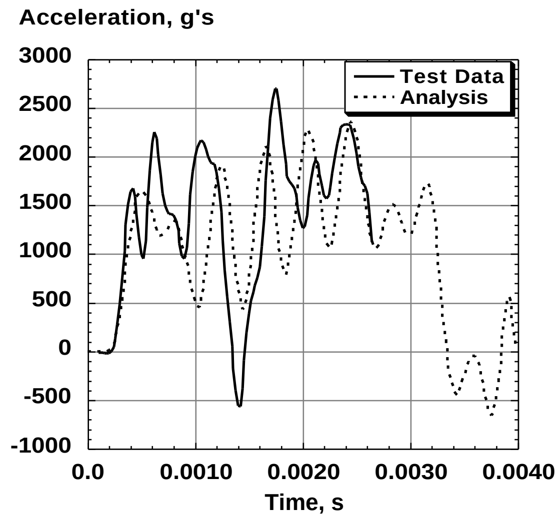

Figure 13 – Predicted acceleration versus experimental acceleration of the impact test ring.

The total crush stoke predicted by the analysis is 0.054 m or 80% of the available crush distance. The simulation showed that maximum crush occurred at approximately 0.002 s after initial impact. The predicted crush compared favorably with that measured from the post-test impact test article, which had a crush stroke of 0.056 m or 82% of the available crush distance. Model deformation as calculated by the finite element simulation is shown in Figure 14. A photograph of the test article after impact is shown in Figure 15.

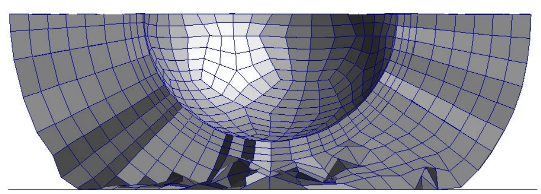

Figure 14 – Cross section of deformed impact structure at maximum crush at 0.002 s.

Figure 15 – Bisected cellular structure, post-test.

Impact Simulation #3

The third impact test specimen included a spherical shaped, full scale OS with its own internal data acquisition system. Hardware representing the current dimensions and weight of the OS were delivered from NASA's Jet Propulsion Laboratory (JPL) for insertion into the cellular structure. The OS had an outside diameter of 0.155 m and had a mass of 3.774 kg. In order to achieve an impact velocity representative of terminal velocity, additional and stiffer bungee cords were used in the bungee accelerator. This change allowed for the impact velocity of the impact test specimen to be 40.4 m/s. One accelerometer was mounted on top of the OS to capture the OS acceleration trace. Two accelerometers were located on the impact test plate to measure the plate acceleration. There was also a JPL supplied accelerometer located within the OS at the approximate CG. Additionally, a low-g accelerometer was used to determine the impact velocity. Digital video was used to capture the test.

Polyurethane foam was replaced by carbon foam, which is better suited for space applications due to low out-gassing, a higher stiffness to weight ratio, and structural integrity at higher reentry temperatures. The amount of Kevlar that was wrapped around the outside of the cellular structure was increased to approximately 0.002083 m. The graphite-epoxy laminate that lined the inner surface of the cellular structure was also increased to 8 layers, or 0.002032 m. The thickness of the cell walls were increased by adding additional layers of graphite-epoxy and Kevlar, providing 18 layers for each cell wall, or 0.004572 m. The thickness of the CV was approximately 0.0048 m. The mass of the cellular structure, CV, OS, and the impact test plate was 14.22 kg. The cellular structure had an outside diameter of 0.308 m and an inside diameter of 0.171 m.

Finite Element Model

A detailed model was constructed to represent the more complex impact test specimen. The cellular structure and impact surface was modeled in the same manner as in the previous impact specimens. The polyurethane foam from previous tests was replaced with carbon foam, which was modeled as a FOAM1 material. A piecewise linear stress strain curve for the carbon foam was constructed using crush test data provided from penetration tests. The density ρ of the carbon foam was 57.656 kg/m^3, and the bulk modulus G was 4.6x10^7 Pa. Additional layers of graphite, graphite-epoxy, and Kevlar were added to the concept resulting in a greater thickness for most of the shell elements. The 4-node shell elements that represented the graphite-epoxy/Kevlar cell walls, the Kevlar outer skin, and the graphite-epoxy inner surface were increased in thickness to match the impact test specimen.

In addition to these modified components, an OS and impact test plate were geometrically modeled and added to the baseline finite element model. A complete detailed model is shown in Figure 16. A cross section of the finite element model is shown in Figure 17.

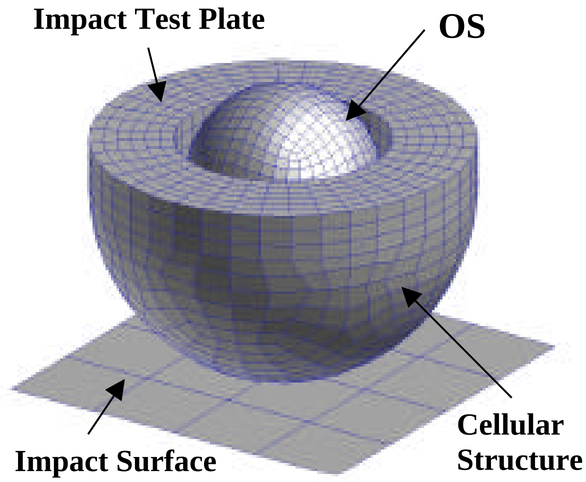

Figure 16 –Components of detailed finite element model.

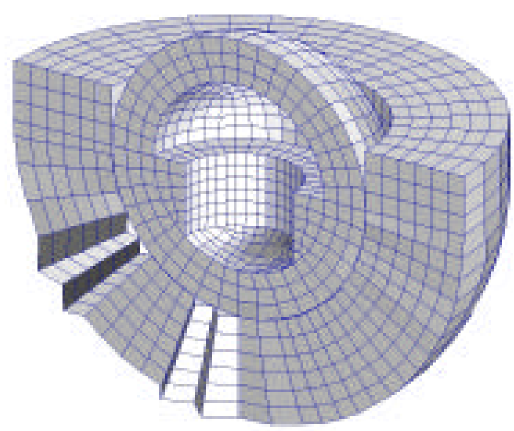

Figure 17 – Cross-section of detailed finite element model.

The complete OS was modeled using 3,528 grid points and 3,560 elements. Of those 3,560 elements, the inner surface of the OS was modeled with 840 4-node Lagrangian shell elements. The shell elements were used to represent the inner surface of the OS and were modeled as a rigid body. The rigid body represents the titanium canister that is used to store the collected samples over the course of the mission. For this impact test, the mass of the titanium canister within the OS was 1.76 kg. Also defined on the MATRIG material card was the impact velocity for this test, 40.4 m/s.

Surrounding the rigid shells are the 2,720 8-node, single integration point, Lagrangian solid elements that represent the foam used in the OS. The material model for these elements is an isotropic, elastoplastic material model with a yield stress defined to allow for plastic deformation. The density ρ of the foam used in the OS was 460 kg/m^3, the Young's modulus E was 4.38×10^8 Pa, the Poisson's ratio ν was 0.334, and the yield stress was 1.54×10^7 Pa.

The impact test plate was modeled using 871 nodes and 720 8-node Lagrangian solid elements. An isotropic material model was used to define the impact test plate. For this impact test, the mass that was added to the top of the cellular structure was 4.25 kg, which resulted in a density ρ of the impact test plate of 4,630 kg/m^3. The Young's modulus E of the impact test plate was 2.72×10^{10} Pa, the Poisson's ratio ν was 0.3, and the yield stress was 2.68×10^8 Pa.

The Two Body Interaction

The CV provides the coupling between the OS and the cellular structure. The measured accelerations at the top of the OS and on the top surface of the impact test plate above the cellular structure and OS are shown in Figure 18. Note that there is a time delay of approximately 0.0005 s from the beginning of the plate/cellular structure acceleration to the initiation of the OS acceleration. This delay is due to the highly nonlinear CV material, which fills the gap between the OS and the cellular structure. In addition, the relatively large spikes at the end of the traces of the impact test plate and the OS are due to impact of the OS with the impact test plate. Consequently, to evaluate the actual dynamic response of the cellular structure and to remove the interaction between the two bodies, the following equation was developed,

$$F(t) = A_{CS}M_{CS} + A_{OS}M_{OS} = M_{TOTAL}A_{SYS}$$

Where F(t) is the crush force of the cellular structure, A_{CS} is the measured acceleration of the combined plate and cellular structure, A_{OS} is the measured acceleration of the OS, and A_{SYS} is the system acceleration (or the acceleration at the cg of the system).

Solving for the system acceleration:

$$A_{SYS} = (A_{CS}M_{CS} + A_{OS}M_{OS}) / M_{TOTAL}$$

For the equation to apply, it was assumed that the impact test plate is rigidly attached to the cellular structure and that the mass of the crushed portion of the sphere is small compared with the total mass. By using the system acceleration, the peaks due to the two separate bodies are eliminated (see Figure 18), and thus a better representation of the behavior of the cellular structure for a "perfect CV coupling" is obtained. Consequently, the CV was incorporated in the OS model by coupling the OS to the cellular structure with equivalent nodes.

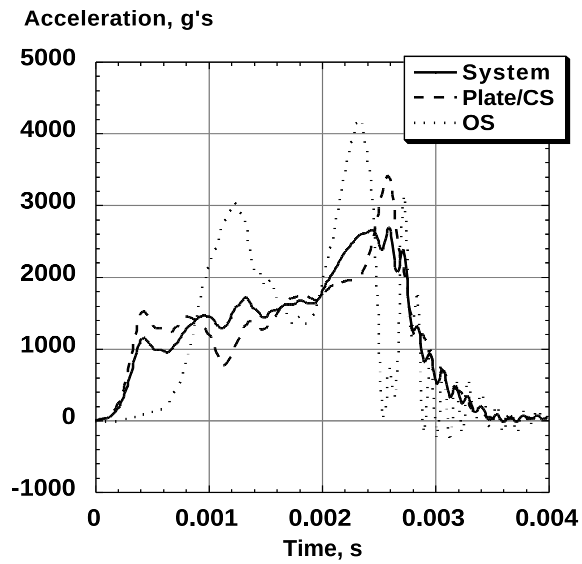

Figure 18 – Measured accelerations of the OS, plate/CS, and the calculated system acceleration.

Numerical Results and Discussion

The simulation was run for 0.004 s to ensure that the complete acceleration pulse was calculated. The maximum time step of the detailed model was 0.65 microseconds, and the model begins to rebound at approximately 0.0035 s. A total run time of one CPU hour was required to compute the 0.004 s impact scenario.

The analysis predicted a peak system acceleration of approximately 2,600 g's, which occurred at approximately 0.0025 s into the impact simulation. This result compares well with the measured peak system acceleration of 2,700 g's. Overall, the simulation accurately predicted the shape, magnitude, and duration of the measured system acceleration pulse, as shown in Figure 19.

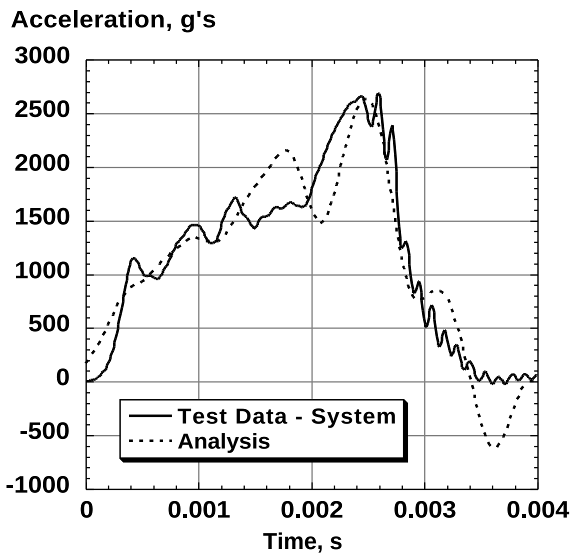

Figure 19 - Comparison of numerical and measured system acceleration.

The total cellular structure crush stroke of the finite element simulation was 0.063 m, or approximately 90% of the available crush distance. The crush of this test specimen was difficult to measure post-test, as the deformed cell walls sprung back, and the outer skin fold lines were not as defined as in the previous tests. However, the crush was estimated to be approximately 85%, or 0.058 m. A deformed plot of the finite element model showing the maximum stroke is shown in Figure 20. A photograph of the cellular structure after the impact is shown in Figure 21.

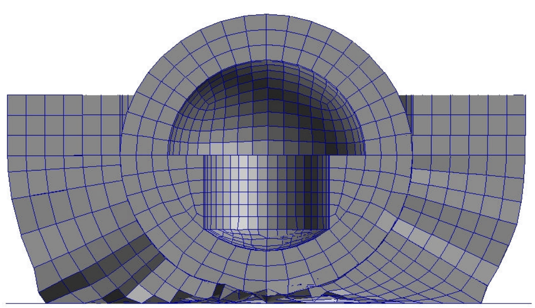
Figure 20 – Deformed plot from finite element simulation showing max crush at 0.003 s


Figure 21 – Photograph of cellular structure cross section, post test.

Summary

Nonlinear transient dynamic finite element simulations were performed using MSC.Dytran to aid in the design of energy absorbing concepts for highly reliable, passive, Earth Entry Vehicles (EEV) that can survive an Earth impact without the aid of a parachute. The current EEV concept uses an energy absorbing impact sphere inside the vehicle to ensure containment and to limit the acceleration of space exploration samples within the Orbiting Sample (OS) container. The energy absorbing impact sphere is a composite cellular structure composed of hybrid graphite-epoxy/Kevlar cell walls filled with energy absorbing carbon foam.

Impact tests of prototype cellular structures were performed for velocities of 32, 35, and 40 m/s at the Impact Dynamics Research Facility (IDRF) at NASA Langley Research Center. For each impact test, a finite element model was created to simulate the event. Numerical data generated from impact simulations created using MSC.Dytran compared favorably with the impact test results for the three cellular structures. The analysis predicted the shape, peak, and duration of the measured acceleration, plus the total crush, quite well.

The material model used to represent the foam and the proper failure criteria for the cell walls were critical in predicting the impact loads on the OS and cellular structure. It was determined that a FOAM1 model for the foam and a 20% failure strain criteria for the cell walls gave an accurate prediction of the acceleration pulse for this range of velocities.

When the cellular structure and OS are modeled separately, the acceleration of each component is complicated due to a two-body interaction. To make meaningful comparisons between analysis and test, a mass-weighted system acceleration of the cellular structure and OS was used. This comparison allowed the acceleration response of the impact test specimen to be determined for a "near perfect" coupling between the OS and cellular structure.

Analytical models have proved useful as a tool for design and analysis of the cellular structure. In addition, parametric studies of off-nominal impact conditions can be simulated at minimal costs to increase the level of confidence of the concept.

Acknowledgement

The authors wish to thank Dr. Robert A. Mitcheltree of NASA Langley Research Center for overall program direction. The research performed by the first author was sponsored by NASA Langley Research Center under NASA Cooperative Agreement NCC1-331 and is a portion of what will be included in a Master of Science Thesis with The George Washington University. The work performed by the last author was sponsored by NASA Langley Research Center under GSA Contract No. GS-35F-4503G, Task 1418. This support is gratefully acknowledged.

References

1. Mitcheltree, R., Braun, R., Hughes, S., Simonsen, L., "Earth Entry Vehicle for Mars Sample Return," IAF Paper 00-Q.3.04, Oct. 2000.

2. Fasanella, E. L., Jones, Y. T., Knight, Jr., N. F., and Kellas, S., "Low Velocity Earth-Penetration Test and Analysis", AIAA paper 2001-1288, 42nd Structures, Structural Dynamics, and Materials Conference, Seattle, WA, April 2001.

3. MSC.Software Corporation, "MSC.Dytran Version 4.7 User's Manual," Volumes 1 and 2, Los Angeles, California, 1999.

4. Vaughan, V. L. and Alfaro-Bou, E., "Impact Dynamics Research Facility for Full-Scale Airccraft Crash Testing," NASA TN D-8179, April 1976.